

# **PERFORMANCE EVALUATION OF TURBULENCE MODELS FOR FLOW SIMULATION OF DOUBLE SUCTION CENTRIFUGAL PUMP**

**Shyam Narayan Shukla**

Corporate Research and Engineering Development, Kirloskar Brothers Limited, Pune, India

**Ruchi Khare**

Maulana Azad National Institute of Technology, Bhopal, India.

**Vishnu Prasad**

Maulana Azad National Institute of Technology, Bhopal, India.

## **ABSTRACT**

*The computational fluid dynamics (CFD) is a cost-effective tool for derivation of performance characteristics of pumps. The turbulent flow comprises the long range of turbulence scales and this needs very fine grid. Turbulence models are used to capture the effect of turbulence and hence play vital role on accuracy of flow simulation. The selection of turbulence model requires understanding of physics of flow domain and validation with experimental results. In present work, the numerical flow simulation of double suction horizontal split case centrifugal pump has been done at different operating points to derive the performance and flow characteristics of pump. The numerically derived performance characteristics are compared with characteristics obtained from physical pump test to select the turbulence model best suited for these types of pumps. It is seen that head, total loss and efficiency at duty point obtained by  $\kappa-\epsilon$  turbulence model bears close comparison to experimental values.*

**Key words:** double suction, horizontal split case, turbulence models, pump efficiency, head coefficient.

**Cite this Article:** Shyam Narayan Shukla, Ruchi Khare and Vishnu Prasad, Performance Evaluation of Turbulence Models for Flow Simulation of Double Suction Centrifugal Pump. *International Journal of Civil Engineering and Technology*, 7(6), 2016, pp.01 – 10.  
<http://www.iaeme.com/IJCIET/issues.asp?JType=IJCIET&VType=7&IType=6>

---

## **1. INTRODUCTION**

The centrifugal pumps have very wide application in water supply and irrigation. The flow passage of centrifugal pump is curved and consists of solid rotating impeller having curved blades. This makes flow very complex inside the pump and designer has to make some assumptions to simplify the design and hence it is required to assess the pump performance. The physical model test of pump in laboratory is the conventional approach for assessment of pump performance which is costly and time consuming. Computational Fluid Dynamics (CFD) is emerging as a cost effective tool for flow simulation in pumps for obtaining detailed flow behavior and derivation of pump performance. Reynolds Averaged Navier Stokes

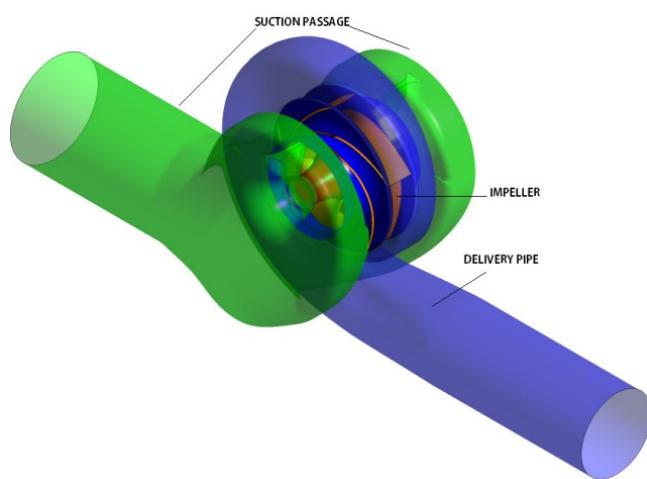
(RANS) equations are solved using numerical techniques like finite control volume method to obtain pressure and velocity distributions in flow domain [Shah S.R. et al, (2013) [2]]. The most of flow simulations has been carried out using eddy viscosity turbulence models.

There are numerous commercial CFD codes are available and used for flow simulation. The accuracy of numerical flow simulation by commercial software depends on many parameters and selection of these parameters requires experience and the validation of CFD results with physical model test results. CFD has been used for design improvement of pump by changing geometry and number of impeller vanes. [Houlin et al (2010) [3], Chakraborty et al, (2012) [4], Bacharoudis et al (2008) [5], Rajmane and Kallurkar (2015)[6]].

The turbulence models used in simulation greatly affect the accuracy as it is used to capture the boundary layer near boundaries for estimation of frictional losses in relatively course grid. The turbulence model suitable for a particular geometry may be unrealistic for other geometries and flow conditions [Durbin et al, (2001) [8]]. The accuracy and reliability of turbulence models can be improved by their modifications [Elkhoury, (2007) [9]]. Hence it is very important to identify the turbulence model which is best suited for a particular application. In present work, the flow simulation in existing double suction impeller with horizontal split case centrifugal pump has been carried out at different operating conditions taking different turbulence models. The pump flow and performance characteristics are derived using commercial CFD code Ansys CFX. These characteristics are compared with those derived from experiment result to select the turbulence model best suited for simulation of these pumps. The flow parameters and losses are also computed and presented in graphical form for different turbulence models for comparison.

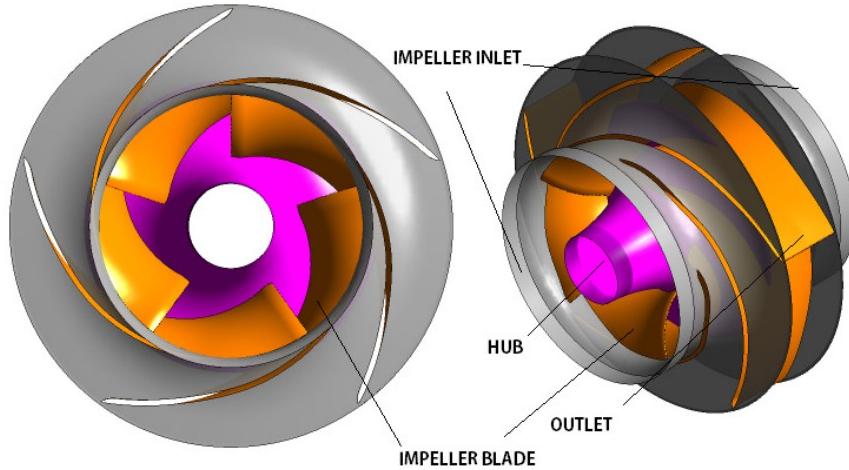
## 2. PUMP GEOMETRY AND MESH GENERATION

The pump design head and discharge coefficient are 0.433 and 0.223 respectively with specific speed NUSUS 1710. The geometry of double suction horizontally split casing pump consists of suction passage divided into two halves, double suction, 5 vane impeller and a volute casing. Out of these three components, impeller is rotating and other two are stationary. Hence geometric modeling of each component is done separately and connected through proper interfaces. The pump geometry for analysis is shown in Figure1. The side and front views of double suction impeller are shown in Figure 2.



**Figure 1** Pump Geometry

The geometry of suction passage, impeller and delivery casing was prepared using Pro Engineer solid modeler. The assembly of pump parts imported to ANSYS ICEM CFD15 for geometry cleaning and meshing.



**Figure 2** Side and front view of impeller

The unstructured hybrid tetrahedral mesh is generated in flow domain except fine prismatic layers near the surface of impeller vanes for proper resolution of the boundary layer. Grid independence study has been performed to ensure appropriateness of grid size and  $Y^+$  value. There are 300000, 1120000 and 200000 nodes in suction passage, impeller and delivery casing respectively.

### 3. BOUNDARY CONDITIONS

The outer enclosing surfaces of all components are taken as walls and no slip condition with appropriate wall roughness was assigned. Average total pressure at suction inlet and Mass flow normal to boundary at casing outlet, are specified as inlet and outlet boundary conditions. The interface between rotating and stationary components is given as frozen rotor. As the frozen rotor interface passes data without averaging them and therefore relative position of impeller vanes with respect to volute tongue remains same during solution. In actual, the relative position of impeller changes continuously with delivery casing. The rotational speed to impeller is specified in rotating frame of reference, while suction and volute regions are kept as stationary. The flow is assumed to be steady in rotating frame of reference. The water at the normal temperature is modeled as incompressible and Newtonian with constant density and viscosity. The scalable Log-law was used near wall.

### 4. TURBULENCE MODELS

The turbulence models are used to capture boundary layer effect in flow simulation with relatively coarse mesh as compared to Direct Numerical Simulation (DNS) scheme. The turbulence models are:

- $\kappa-\epsilon$  model
- Shear stress transport (SST)
- Renormalized group  $\kappa-\epsilon$  (RNG  $\kappa-\epsilon$ )
- $\kappa-\omega$  turbulence model
- Speziale, Sarkar and Gatski Reynolds stress model (SSG RS)
- Speziale, Sarkar and Gatski, Baseline Reynolds stress model (SSG BSL RS)
- Speziale, Sarkar and Gatski Zero equation model (SSG ZE)

Out of above models, first four are eddy-viscosity models. Both the  $\kappa-\epsilon$ ,  $\kappa-\omega$  or the blends of these turbulence models relate the Reynolds stress and turbulent flux terms to the mean flow variables. The gradient diffusion hypothesis is used to relate the Reynolds stresses with the mean velocity gradients and the turbulent viscosity. In Reynolds stress model, the Reynolds stresses are computed directly using six additional transport equations which take more computational time. The flow simulation with all above models has been carried at duty point and it is observed that values of flow parameters obtained from SSG

models are much deviating from experimental data. Hence eddy-viscosity models are considered in the present study.

## 5. COMPUTATION OF PARAMETERS

The numerical simulation gives values of pressure and velocity distribution in flow domain of pump. The following local and global flow and loss parameters are computed in non-dimensional form as:

$$\text{Head developed} \quad H = \frac{TP_o - TP_l}{\gamma} \quad (1)$$

$$\text{Flow coefficient} \quad Q_s = \frac{Q}{\omega r_2^3} \quad (2)$$

$$\text{Head coefficient} \quad \psi = \frac{gH}{\omega^2 r_2^2} \quad (3)$$

$$\text{Total head loss} \quad H_{HL} = H_s + H_I + H_c \quad (4)$$

$$\text{Relative loss in suction} \quad \xi_s = \frac{H_s}{H + H_{HL}} \times 100 \quad (5)$$

$$\text{Relative loss in impeller} \quad \xi_I = \frac{H_I}{H + H_{HL}} \times 100 \quad (6)$$

$$\text{Relative loss in casing} \quad \xi_c = \frac{H_c}{H + H_{HL}} \times 100 \quad (7)$$

$$\text{Relative total head loss} \quad \xi_{HL} = \frac{H_{HL}}{H + H_{HL}} \times 100 \quad (8)$$

$$\text{Hydraulic efficiency} \quad \eta_{hyd} = \frac{H}{H + H_{HL}} \times 100 \quad (9)$$

$$\text{Overall efficiency} \quad \eta_o = \eta_{hyd} \times \eta_{vol} \times \eta_{mech} \quad (10)$$

$$\text{Pressure Coefficient} \quad C_p = \frac{P}{\frac{1}{2} \rho U_2^2} \quad (11)$$

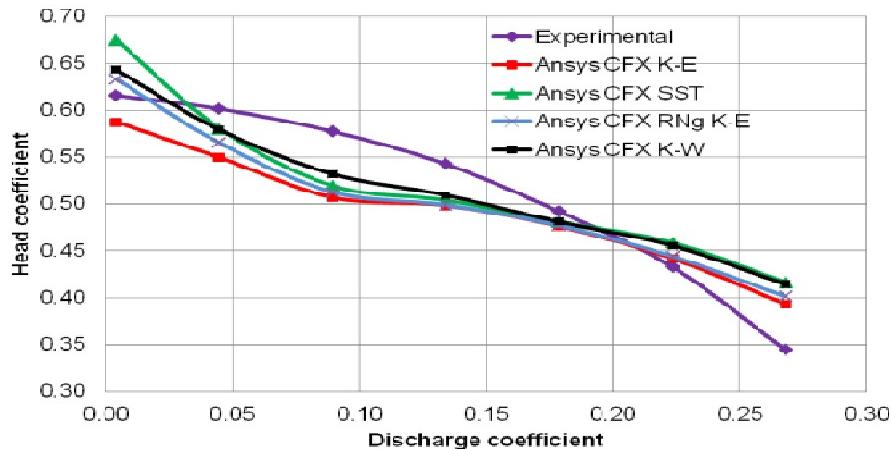
## 6. RESULTS AND DISCUSSIONS

The flow simulation is carried out for seven different discharges varying from 2% to 120 % of design discharge at constant rotational speed of pump for four turbulence models. The values of discharge coefficient for 2%, 20%, 40%, 60%, 80%, 100% and 120% are 0.004, 0.045, 0.089, 0.134, 0.179, 0.223 and 0.268 respectively. As the simulation at zero discharge is not possible, simulation is carried out for 2% discharge.

In experimental results, the measured efficiency (overall efficiency) includes volumetric and mechanical losses which are not estimated in experiment. It is also not possible to estimate these losses from CFD. Hence the volumetric and mechanical losses corresponding to the specific speed and discharge at rated conditions were estimated from the graphs given by Karassik [1] and assumed to be same at all discharges. Then experimental hydraulic efficiency is calculated by equation (10). The computed local and global parameters from the flow simulation results are presented in qualitative and quantitative form. The quantitative values of global parameters i.e. head and discharge coefficient and efficiency obtained from

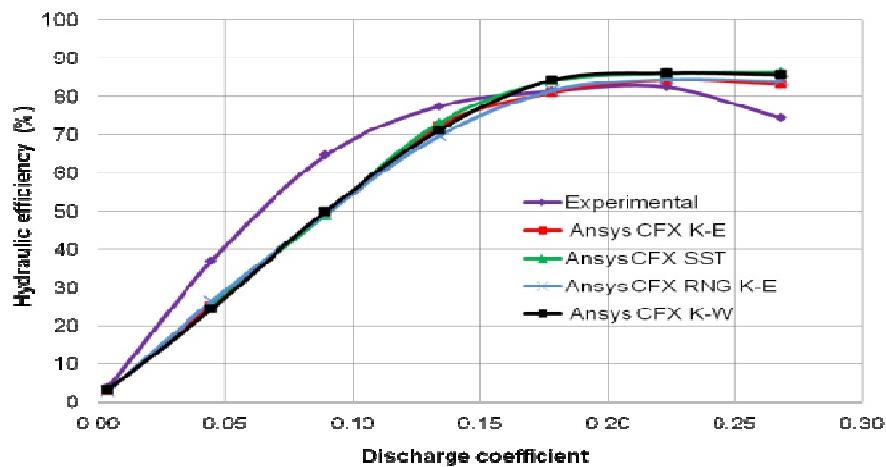
different turbulence models are compared with the experimental data to assess the suitability of turbulence model for this pump.

To calculate the pump efficiency from simulation results, the hydraulic losses are estimated in suction passage, impeller and volute casing. These losses combined together form the basis for hydraulic efficiency of the pump. The difference in relative total pressure at entry and exit of the impeller gives the loss in a rotating blade passage. The ratio of head developed by pump to pump head plus losses gives the hydraulic efficiency of pump.

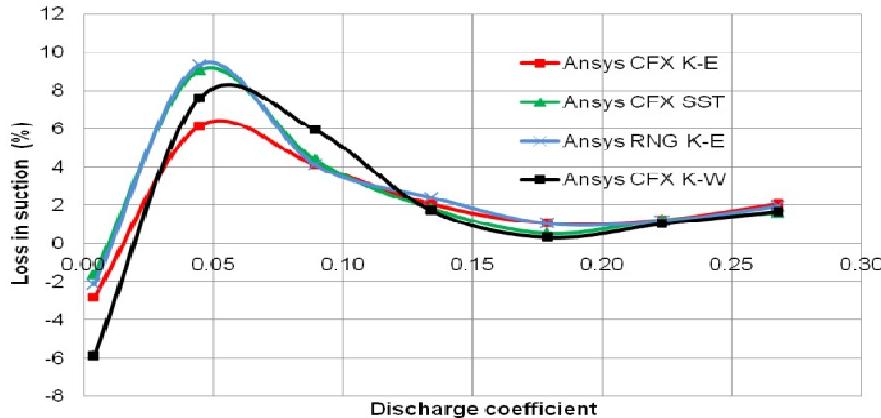


**Figure 3** Variation of head

It is seen from figure 3 and figure 4 that variation patterns of head, hydraulic efficiency and their values near design condition are closely matching with experimental results but there are large variations at off design condition. It is observed from the figure 3 that at load conditions 20-70% load, head and efficiency obtained from CFD are less than experimental value while at overload (120%), their values from CFD are more than experimental. This indicates that the losses are not properly accounted in CFD at off-design conditions due to secondary flows. The head developed is found minimum from  $\kappa - \epsilon$  model while it is maximum from SST turbulence model nearly at all operating conditions. The hydraulic efficiency is nearly same up to 60% discharge from all turbulence models and  $\kappa - \epsilon$  model gives efficiency close to experimental near design point as seen from figure 4.



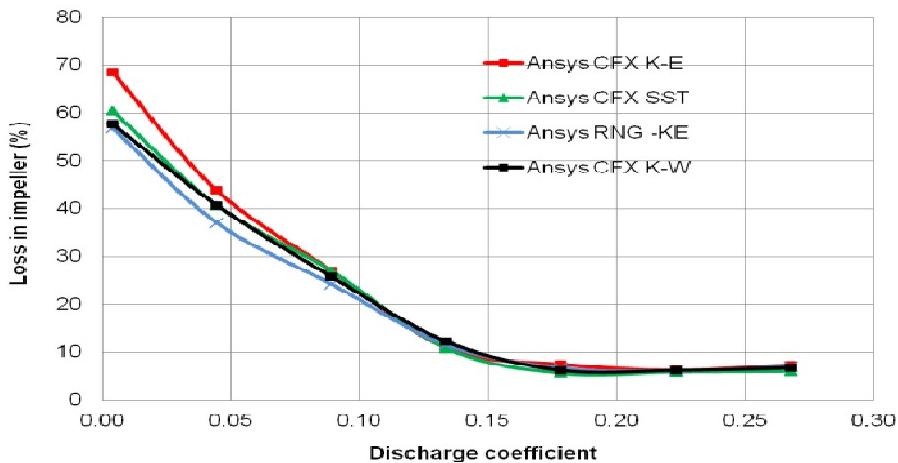
**Figure 4** Variation of hydraulic efficiency



**Figure 5** Variation of loss in suction

The hydraulic losses are computed in each component of pump from CFD results at different discharge and normalized with the total head developed including losses. The variations of hydraulic losses in different components and total loss in figure 5, figure 6, figure 7 and figure 8 indicate that the more loss occurs at the lower discharge and it decreases as the discharge increases up to design point. The losses obtained at design discharge in different components are nearly same from all turbulence models.

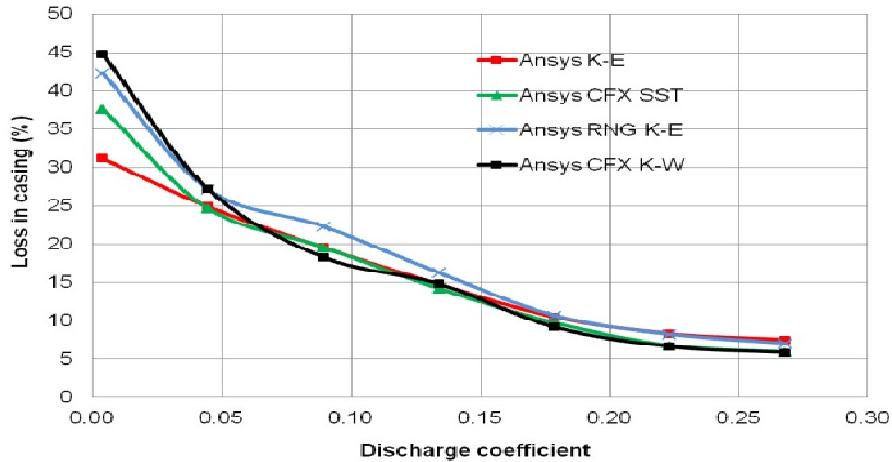
There is drastic drop in loss as discharge increases from zero to 60%. The loss in suction at 2% discharge comes out to be negative in figure 5 which indicates that at very low discharge, loss prediction by CFD is not possible in suction passage. This may be due to highly circulatory flow at suction. It is seen from figure 3 that RNG  $\kappa$ - $\varepsilon$  gives maximum suction loss while  $\kappa$  –  $\varepsilon$  gives minimum loss.



**Figure 6** Variation of loss in impeller

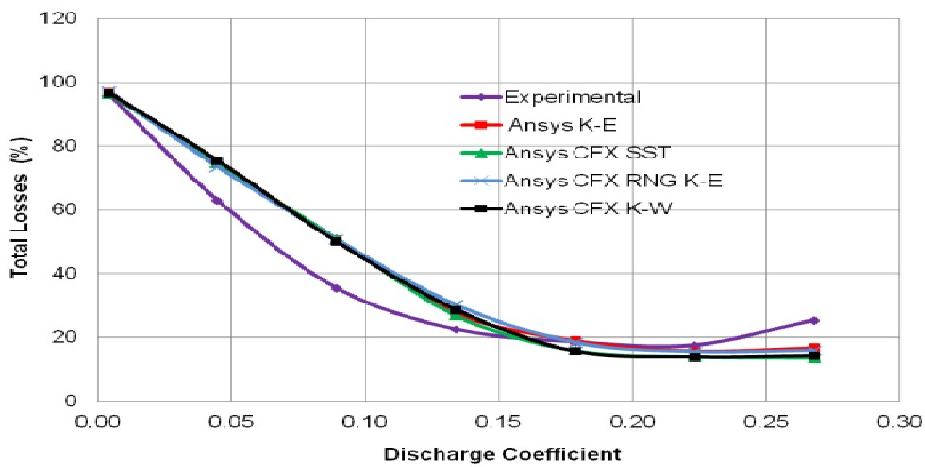
The loss variation in impeller in figure 6 indicates that loss decreases as the discharge increases and becomes minimum at design discharge and again slightly increases. The loss prediction in impeller is more with  $\kappa$  –  $\varepsilon$  as compared to other models.

The loss in casing is also found to decrease with increase in discharge in figure 7. It is seen that  $\kappa$  –  $\varepsilon$  gives less loss at lower discharge and more loss at higher discharge while  $\kappa$ - $\omega$  gives more loss at lower discharge and less loss at higher discharge.



**Figure 7** Variation of loss in casing

The total loss obtained from different turbulent models is compared with the experimental values in figure 8. It is observed that total loss obtained in experiment is much lower than that obtained from CFD for 2% to 70% load while it is more at overload condition and hence it is just reverse of hydraulic efficiency variation. The total loss variation up to 70% load condition from different turbulent models is nearly same but loss values obtained from different models differ after 70% load. The minimum loss is predicted by  $\kappa - \omega$  model. The total loss value closely matches with experimental value for RNG  $\kappa - \varepsilon$  and  $\kappa - \varepsilon$  models near design discharge.



**Figure 8** Variation of total head loss in pump

The mass averaged values of flow parameters in non-dimensional form are computed for 60%, 100% and 120% discharge at inlet and outlet of impeller to study the effect of turbulence models on these parameters. The static pressure coefficient values in Table-1 indicate that pressure increases from inlet to outlet. The pressure at inlet and outlet is nearly independent of turbulence models. Secondly the pressure rise from inlet and outlet is decreasing with increase in discharge. It is seen that the velocity components in Table-2, Table-3 and Table-4 are affected by turbulence model at part load operation i.e. 60% load but at design and overload condition, their values are nearly same and independent of turbulence model. The absolute velocity is increasing at outlet with increase in discharge. The variations at part load condition for different turbulence models may be due to non-uniform distribution of flow across the sections. The variations of flow parameters from inlet to outlet confirm the flow characteristics of pump.

**Table 1** Variation of static pressure coefficient at different load condition

Turbulence model	60% load			100% load			120% load		
	Inlet1	Inlet2	Outlet	Inlet1	Inlet2	Outlet	Inlet1	Inlet2	Outlet
$\kappa - \varepsilon$	0.13	0.13	0.87	0.12	0.12	0.79	0.10	0.10	0.72
SST	0.13	0.13	0.85	0.11	0.12	0.82	0.11	0.10	0.75
RNG $\kappa - \varepsilon$	0.12	0.11	0.85	0.12	0.12	0.79	0.09	0.10	0.73
$\kappa - \omega$	0.13	0.13	0.87	0.12	0.12	0.80	0.11	0.10	0.75

Some whirl is observed at both the inlets of pump hence the assumption of no swirling at inlet of horizontal split case pump is not valid (Table-4). This is because of turning of flow by 90° in suction passage before its entry into pump impeller which creates pre-whirl in flow. The whirl velocity at outlet is more than inlet and it is nearly same for all turbulence models. The whirl is increasing from inlet to outlet because of energy imparted by rotating impeller. The rise in whirl velocity decreases with increase in discharge.

**Table 2** Normalized absolute velocity ( $C/U_2$ ) at different load condition

Turbulence model	60% load			100% load			120% load		
	Inlet1	Inlet2	Outlet	Inlet1	Inlet2	Outlet	Inlet1	Inlet2	Outlet
$\kappa - \varepsilon$	0.13	0.13	0.69	0.18	0.18	0.56	0.21	0.22	0.52
SST	0.11	0.14	0.71	0.19	0.19	0.55	0.21	0.20	0.52
RNG $\kappa - \varepsilon$	0.16	0.15	0.74	0.19	0.19	0.56	0.22	0.22	0.52
$\kappa - \omega$	0.14	0.14	0.72	0.19	0.19	0.56	0.21	0.20	0.52

**Table 3** Normalized relative velocity ( $W/U_2$ ) at different load condition

Turbulence model	60% load			100% load			120% load		
	Inlet1	Inlet2	Outlet	Inlet1	Inlet2	Outlet	Inlet1	Inlet2	Outlet
$\kappa - \varepsilon$	0.40	0.37	0.42	0.43	0.43	0.51	0.43	0.43	0.55
SST	0.36	0.40	0.43	0.46	0.46	0.52	0.45	0.45	0.55
RNG $\kappa - \varepsilon$	0.36	0.36	0.44	0.43	0.43	0.51	0.43	0.43	0.55
$\kappa - \omega$	0.41	0.43	0.42	0.44	0.45	0.52	0.45	0.45	0.55

**Table 4** Normalized whirl velocity ( $C_U/U_2$ ) at different load condition

Turbulence model	60% load			100% load			120% load		
	Inlet1	Inlet2	Outlet	Inlet1	Inlet2	Outlet	Inlet1	Inlet2	Outlet
$\kappa - \epsilon$	0.03	0.01	0.65	0.05	0.05	0.53	0.07	0.07	0.48
SST	0.004	0.01	0.66	0.03	0.02	0.52	0.05	0.05	0.48
RNG $\kappa - \epsilon$	0.01	0.00	0.68	0.05	0.05	0.53	0.07	0.06	0.48
$\kappa - \omega$	0.02	0.02	0.67	0.04	0.04	0.52	0.05	0.05	0.48

## 7. CONCLUSION

It is found that the numerical flow simulation in horizontal split case centrifugal pump is more affected by turbulence models at part load operations and it is nearly independent at design and overload conditions. The global parameters like head and efficiency obtained from  $\kappa-\epsilon$  and RNG  $\kappa-\epsilon$  models are more close to experimental values at design point. The major losses occur in casing and impeller and found to decrease with increases in discharge. The losses are not assessed accurately at part load operation with all eddy viscosity turbulence models and no definite pattern is observed.

The local velocity and pressure are more affected at inlet with different turbulence models and this may be due to variation in their distribution across flow area. Based on present study, it is concluded that  $\kappa-\epsilon$  and RNG  $\kappa-\epsilon$  models can be suitable for pumps near to design point and no turbulence model predict loss accurately at part load conditions. The flow parameters computed at inlet and outlet of impeller confirm the characteristics of pump in all turbulence models.

## 8. NOMENCLATURE

- H - head to be developed by pump
- $H_S$  - head loss at suction side of pump
- $H_I$  - head across impeller of pump
- $H_D$  - head loss at delivery of pump
- $H_{HL}$  - total head loss in pump
- Q - discharge of pump
- g - gravitational acceleration
- $T_{PI}$  - total pressure at pump inlet
- $T_{PO}$  - total pressure at pump outlet
- P - static pressure
- $r_2$  - pump impeller outlet radius
- $\omega$  - rotational speed of impeller
- $\gamma$  - specific weight of water
- $\eta_{pump}$  - pump efficiency
- $\eta_{hyd}$  - hydraulic efficiency
- $\eta_{vol}$  - volumetric efficiency
- $\eta_{mech}$  - mechanical efficiency

## REFERENCE

- [1] Karassik Igor J, Messina Joseph P, Cooper Paul, Heald Charles C., "Pump Hand book" Fourth Edition, 2008, The McGRAW-Hill Publication.
- [2] Shah S.R., Jain S.V., Patel R.N. and Lakheria V.J., "CFD for centrifugal Pump:A review of the state of the art" Procedia Engineering 51 ( 2013 ).pp 715 – 720
- [3] Houlin LIU, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai," Effects of Blade Number on Characteristics of Centrifugal Pump", Chinese Journal Of Mechanical Engineering ,Vol. 23, July , 2010 DOI: 10.3901/CJME.2010,pp 1-6
- [4] ChakrabortySujoy, Pandey K.M., RoyaBidesh, " Numerical Analysis on Effects of Blade Number Variations on Performance of Centrifugal Pumps with Various Rotational Speeds", International Journal of Current Engineering and Technology, Vol.2, No.1 (March 2012) ISSN 2277 - 4106 Research Article
- [5] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris. Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle. The Open Mechanical Engineering Journal, 2008, 2, 75-83.
- [6] RajmaneSatish M, . S. P. Kallurkar," Enhancing Performance Of Domestic Centrifugal Pump By Use Of CFD Tool By Changing Design Parameters", International Journal of Informative & Futuristic Research, Volume 3 Issue 2 October 2015 ,pp 579-590
- [7] Leng, H. F.; Wang, F. J.; Zhang, Z. C.; Yao, Z. F.; Zhou, P. J., "Suppression of secondary flows in a double suction centrifugal pump with different loading distributions", Materials Science and Engineering, Volume 52, Issue 3, article id. 032008 (2013).
- [8] Durbin P.A. and Pettersson R.B.A., Statistical Theory and Modelling of Turbulent Flows, Wiley, New Yoik, 2001.
- [9] Elkhoury M., Assessment and Modifications on One equation Models of turbulence for Wall bounded Flows, J. Fluids Engg., Vol.129, 2007, pp.921-928.
- [10] ANSYS-CFX Users guide, V 14.5, by M/s ANSYS Inc USA (2014).
- [11] Manish Dadhich, Dharmendra Hariyani and Tarun Singh, Flow Simulation (CFD) & Fatigue Analysis (FEA) of a Centrifugal Pump. *International Journal of Mechanical Engineering and Technology (IJMET)*, 3(3), 2012,pp. 67–83.
- [12] Aakash M Bodh and Prof. G.H. Waghmare, Study, Design and Improvement of Pumping System Efficiency of Hydraulic Pneumatic Reciprocating Pump. *International Journal of Mechanical Engineering and Technology (IJMET)*, 7(5), 2016, pp. 127–132.